A SPEEDy user's guide

The input for SPEED is divided in 4 main files:

- *a general header file (SPEED.input) that fixes the fundamental parameters of the analysis*
- *a mesh file (FileName.mesh) in which the grid connectivity is given*
- *a specific file for the problem under examination (FileName.mate) where material properties, boundary conditions and external loads are given.*
- *a list of monitored points (LS.input) for which the solutions is written in output.*

All input files are ASCII text files. Command lines are not ordered, nor structured. A command line is identified by a 7 characters keyword, starting the line. If a line does not start with a recognized sequence of characters is considered a comment. That is, a space at the beginning of a line identifies that line as a comment. No other specific syntax for the comments is given and only minor syntax check is provided at the moment.

MAIN INPUT FILES

SPEED.input: *Header file.*

Filename.mesh: *Mesh description.*

Filename.mate: *Material properties, boundary conditions and source terms description.*

LS.input: *List of monitored points.*

Other secondary files, which may be needed or not, according to the simulation, are:

OTHER INPUT FILES

PG.input: *List of monitored points for Peak Ground Map.*

ALL.out, XYZ.out: *Mesh files for the topography and alluvial basin description (not-honoring approach).*

MLST.input: *File created during the execution of SPEED containing infos about monitored points.*

FACES.input: *File created during the first execution of SPEED containing infos about DG surfaces.*

DGFS.input: *File created during the first execution of SPEED containing infos about DG quadrature nodes.*

In the following we report a list of recognized keyword to be used within the MAIN INPUT FILES.

SPEED.input

List of keywords:

GRIDFILE FILENAME
FILENAME.mesh contains the description of the mesh (see Filename.mesh section)

MATFILE FILENAME
FILENAME.mate contains the description of the material properties, interface and boundary conditions and external source terms (see Filename.mate section)

MPIFILE DIRNAME*
*DIRNAME: Name of the directory in which you want to store files *.mpi during the execution of SPEED. If absent files *.mpi will be stored in the working directory.*
**WARNING: If the keyword is present, then the folder DIRNAME must be created in the working directory before running SPEED.*

MONFILE DIRNAME*
DIRNAME: Name of the directory in which you want to store files MONITOR. during the execution of SPEED. If absent files MONITOR.* will be stored in the working directory.*
**WARNING: If the keyword is present, then the folder DIRNAME must be created in the working directory before running SPEED.*

BKPFILE DIRNAME*
*DIRNAME: Name of the directory in which you want to store files backup files *.out (for the restart) during the execution of SPEED. If absent files *.out will be stored in the working directory.*
**WARNING: If the keyword is present, then the folder DIRNAME must be created in the working directory before running SPEED.*

DGMETHOD VAL
Select the DG method used, in particular:
VAL: 1 Non-symmetric IP, 0 Incomplete IP, -1 Symmetric IP (recommended)

PENALIZC VAL
VAL: Penalty value for DG approximations. VAL > 100 is recommended to prevent instability.

STARTIME T0
T0: Initial time for the simulation. Default T0 = 0.

STOPTIME TF
TF: Final time for the simulation.

TIMESTEP DELTAT
DELTAT: Time integration step adopted.

TIMEFIXE DELTAT_FIX
DELTAT_FIX: “yes” or “not”. Fixed (or not) time integration step.

TMONITOR TMON
*Write output results at any time that is a multiple of TMON*TIMESTEP.*

TIMESCHM *SCHEME ORDER STAGES*
Use this keyword for selecting Runge-Kutta time integration scheme.
Set SCHEME=RUNGEKUTTA, and the couple ORDER STAGE equal to 2 2, 3 3, or 4 4.

TESTMODE *TEST*
Use this keyword with TEST=1, for using a special test case (for which the analytical solution is known and you want to carry out the error analysis)

TIMERR *TERR*
TERR is the instant of time in which you want to compute the error between the approximated solution and the analytical one. Use this keyword coupled with TESTMODE.

SNAPSHOT SNAP
SNAP: Instant in which you want to make the backup of the solution. Useful for a restart of the simulation.

OPTIOUT DIS VEL ACC STRESS STRAIN ROT FORMAT DATA
Options for the output
DIS: 1(0) save (do not save) the displacement field
VEL: 1(0) save (do not save) the velocity field
ACC: 1(0) save (do not save) the acceleration field
STRESS: 1(0) save (do not save) the stress tensor
STRAIN: 1(0) save (do not save) the strain tensor
ROT: 1(0) save (do not save) the rotational tensor
FORMAT: DUMMY
DATA: DUMMY

MLST DEPTH VAL
DEPTH: Depth for searching the monitored points
VAL: 1(0) Read(Write) MLST.input in the working directory

PGDM DEPTH TMONPG ROTANG VAL
DEPTH: Depth for searching the monitored points for Peak Ground Maps
*TMONPG: Write Peak Ground Map at each time that is multiple of TMONPG*TIMESTEP*
ROTANG: DUMMY
VAL: 1(0) Read(Write) PG.input in the working directory

FileName.mate

List of Keywords:

MATE BLOCKID POLDEG RHO LAMBDA MU GAMMA
BLOCKID: Id for the material, defined and exported from the mesh generator.
POLDEG: Polynomial degree adopted for the material
RHO: Mass density
LAMBDA: Lamè coefficient (first modulus)
MU: Lamè coefficient (second modulus)
GAMMA: Damping factor (material dissipative coefficient)

MATN BLOCKID DEGREE NF DEPTH
Assigns non linear elastic properties to a group (BLOCKID) of elements.
BLOCKID: block number defined and exported from the mesh generator
DEGREE: polynomial approximation degree used in the subdomain with label BLOCKID
NF: number of function associated with this material property
DEPTH: depth (with respect to the topography) of the non linear layer.

ABSO FACEID
FACEID: Id for the block, defined and exported from the mesh generator. Absorbing boundary conditions are applied to faces of hexahedral elements.

DGIC FACEID VAL
FACEID: Id for the block, defined and exported from the mesh generator for DG interfaces.
VAL: 1(0) Project(Receive) quadrature nodes
DG interface conditions are applied to faces of hexahedral elements.

CASE NCASE BLOCKID TOL
Assign material properties node by node through a not-honoring strategy. Input files ALL.out and XYZ.out are needed.
NCASE: Id number for particular scenario considered
BLOCKID: Id for the block where the not-honoring strategy is applied
TOL: tolerance for finding spectral nodes

FMAX FREQ
FREQ: Maximum frequency value used in the computation.

FUNC NF TYPE PARAMETERS
Defines a function of time, used to scale applied loads and boundary conditions. Different formulations and function types are possible and are specified with the second number in the syntax.
NF: Id number of the function used for distinguish different functions
TYPE:

- *0 - Unit Constant Load*
- *1 - Ricker "beta" type wavelet*
  Parameters: beta t0
- *2 - Ricker "cos" type wavelet*
  Parameters: beta t0
- *3 - Time function of (Nt,Ft) given values written in FileName*
  Parameters: Nt FileName
- *4 - First derivative of Ricker wavelet*
  Parameters: beta t0
- *6 - Ricker "beta" type wavelet, for seismic moment*
  Parameters: beta t0

12 - Smoothed ramp source: sigmf(t,[a c]) = amp(1/(1+exp(-a*(t-c))))*
Parameters: amp, a, c
13 - Grenoble benchmark (contact developers)
14 - SCEC benchmark (contact developers)
*15 - Explosion: val = ps0 * (1 - (1 + t/tplus)*exp(-alpha*t/tplus));*
Parameters: ps0, tplus, alpha, t0 (time delay)
60 - Function for G/G0 (linear equivalent option, contact developers)
61 - Function for damping (linear equivalent option, contact developers)
99 - Cashima Benchmark (contact developers)
100 - Testmode case (contact developers)

SISM NF BLID XIPO YIPO ZIPO X1 Y1 Z1 X2 Y2 Z2 X3 Y3 Z3 S1 S2 S3 N1 N2 N3 V M T
Assign seismic moment. Fault plane is discretized using triangles. For each of them the load is specified with a specific "SISM" instruction.
NF : Number of time function associated with the load
BLID : Block id number defined and exported from the mesh generator
XIPO, YIPO, ZIPO: Coordinates of ipocenter
Xi,Yi,Zi: Coordinates of the three vertex of the triangle
S1,S2,S3: Cosine director of the slip vector
N1,N2,N3: Cosine director of the normal vector
V: Rupture velocity
M: Value of the seismic moment applied to the triangle
T: Rising time

EXPL NF BLID XIPO YIPO ZIPO X1 Y1 Z1 X2 Y2 Z2 X3 Y3 Z3 S1 S2 S3 N1 N2 N3 V M T
Assigns an explosive source. As for seismic moment, fault plane is discretize using triangles. For each of them the load is specified with a specific "EXPL" instruction.
NF: number of time function associated with the load
BLID: block number defined and exported from the mesh generator
XIPO, YIPO, ZIPO: coordinates of ipocenter
Xi, Yi, Zi: coordinates of the three vertex of the triangle
S1, S2, S3: cosine director of the slip vector
N1, N2, N3: cosine director of the normal vector
V: rupture velocity
M: value of the seismic moment applied to the triangle
T: rising time

PLAX NF BLOCKID VAL
PLAY NF BLOCKID VAL
PLAZ NF BLOCKID VAL
Defines a plane wave source. This is realized introducing a force time history, able to generate a displacement time history with the Ricker wavelet shape (NF = 4), on a certain surface.
The load along the X, Y or Z direction is specified as the product of a time function of index NF, times a point load of intensity VAL integrated on the element.
NF: number of function associated with this load: use 4
BLOCKID*: block id number defined and exported from the mesh generator*
VAL: value of the applied load

NEUX BLOCKID NF VAL1 VAL2 VAL3 VAL4
NEUY BLOCKID NF VAL1 VAL2 VAL3 VAL4
NEUZ BLOCKID NF VAL1 VAL2 VAL3 VAL4
NEUN BLOCKID NF VAL1 VAL2 VAL3 VAL4
Assigns Neumann boundary conditions to a group (BLOCKID) of surface (quad) elements. The load may be applied in the X, Y, Z or Normal direction. The load is specified as the product of a time function of index NF, times a distributed load varying from VAL1 to VAL4.
BLOCKID block id number defined and exported from the mesh generator
NF: number of function associated with this load
VALi: value of Neumann load at vertex i of the element

DIRX BLOCKID NF VAL1 VAL2 VAL3 VAL4
DIRY BLOCKID NF VAL1 VAL2 VAL3 VAL4
DIRZ BLOCKID NF VAL1 VAL2 VAL3 VAL4
Assigns Dirichlet boundary conditions to a group (BLOCKID) of surface (quad) elements. The load may be applied in the X, Y or Z direction. The load is specified as the product of a time function of index nf, times a distributed load varying from VAL1 to VAL4.
BLOCKID block id number defined and exported from the mesh generator
NF: number of function associated with this load
VALi: value of Dirichlet load at vertex i of the element

PLOX NF X Y Z VAL
PLOY NF X Y Z VAL
PLOZ NF X Y Z VAL
Assigns Neumann type point load to the spectral node nearest to a specified X,Y,Z position. The load along the X, Y or Z direction is specified as the product of a time function of index NF, times a point load of intensity VAL.
NF: number of function associated with this load
X Y Z: coordinates of the desired point of application
VAL: value of the applied load

TLOX NF VEL AMP*
TLOY NF VEL AMP*
TLOZ NF VEL AMP*
Assigns Neumann type point load to the spectral nodes nearest to the ones specified on the file TRAVPOINTS.LOAD. The load along the X, Y or Z direction is specified as the product of a time function of index NF, times a point load of intensity AMP that moves with velocity VEL.
**Not yet tested.*

FORX NF X Y Z VAL
FORY NF X Y Z VAL
FORZ NF X Y Z VAL
Assigns a force (volume) load along the X, Y or Z direction.
NF: number of function associated with this load
X Y Z: coordinates of the desired point of application

VAL: value of the applied load

FORC NF XC YC ZC VAL R1 R2 R3 PHI THETA PSI
Assigns the volume force load : VAL* exp(- || x / R ||^2), R=(R1,R2,R3).
NF: number of function associated with this load
XC YC ZC: coordinates of the desired point of application
VAL: amplitude of the desired force
R1 R2 R3: components of vector R
PHI THETA PSI: polar coordinate representation

PRES NF XC YC ZC VAL R1 R2 R3 PHI THETA PSI
*Assigns a pressure load: -2.0 * VAL * x/(R^2) * exp(- || x / R ||^2), R=(R1,R2,R3).*
NF: number of function associated with this load
XC YC ZC: coordinates of the desired point of application
VAL: amplitude of the desired force
R1 R2 R3: components of vector R
PHI THETA PSI: polar coordinate representation

SHEA NF XC YC ZC VAL R1 R2 R3 PHI THETA PSI
*Assigns a pressure load: (0, -2*x2/(R2^2), 2*x3/(R3^2)) * VAL * exp(- || x / R ||^2), R=(R1,R2,R3).*
NF: number of function associated with this load
XC YC ZC: coordinates of the desired point of application
VAL: Amplitude of the desired force
R1 R2 R3: components of vector R
PHI THETA PSI: polar coordinate representation

TEST NF
NF: number of function used in the TESTMODE case.

FileName.mesh (ALL.out, XYZ.out)

Mesh file created with Cubit program and then converted in the following format

First line: *total_number_of_nodes total_number_of_elements 0 0 0*
Second line: *ID_node X Y Z*
Third line: *ID_node X Y Z*
Fourth line: *ID_node X Y Z*
.....
.....
.....
(following): *ID_element #_block element_type connectivity nodes*

.....
.....
.....

LS.input

The list of monitors has to be in written in the following format

First line: *Total_number_of_monitors*
*S*econd line: *X* *Y* *Z*
Third line: *X* *Y* *Z*
*F*ourth line: *X* *Y* *Z*
.....
.....
.....
Total_number_of_monitor + 1: *X* *Y* *Z*